> " Femap has built a reputation over the last ten years as a Swiss army knife of FEA, bridging the gap between 3D CAD geometry and simulation "

Time to market is and arguably has always been, the big thing for those involved in product development and manufacturing. It's consistently been shown that those that take a solid, robust design to market earlier gain the benefits of adoption, sales and competitive advantage. With today's technology playing an increasingly critical role in that process, simulation has become an area in which many organizations are concentrating their efforts. By simulating a product's performance and durability early in the development process, as design is still fluid, a better understanding can be gained of how the product is performing both against internal requirements and that of the user.

To assist with getting results back and integrated into the design process, many are now looking to automation to add intelligence to the simulation process. At the same time, compression of time needs to be offset against ensuring that the data you're working on, is valid and can be used effectively and efficiently.

This sets the scene for Siemens PLM Software Femap version 10, which brings automation to the fore, but does so with a series of tools that allow the user to dive in, adapt and refine the mesh and the loading conditions of a study, to ensure that the results are accurate and produced in a timely manner.

Let's dig in and see what it can do.

Al Dean
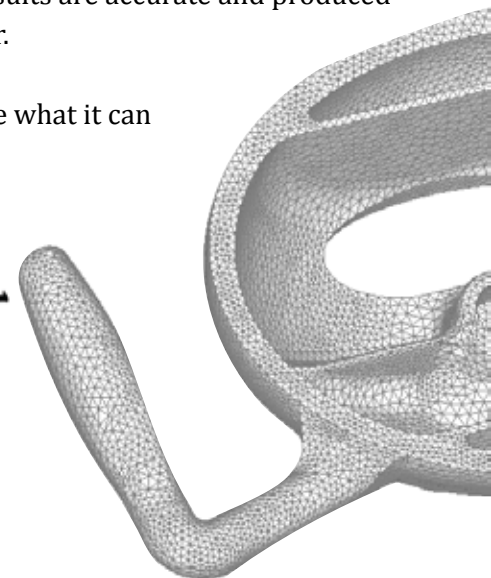
Femap has built a reputation over the last ten years as a Swiss army knife of FEA, bridging the gaps between those systems that author and edit 3D CAD geometry and those systems responsible for carrying out the simulation calculation. It reads a wide variety of both geometry formats and simulation data.
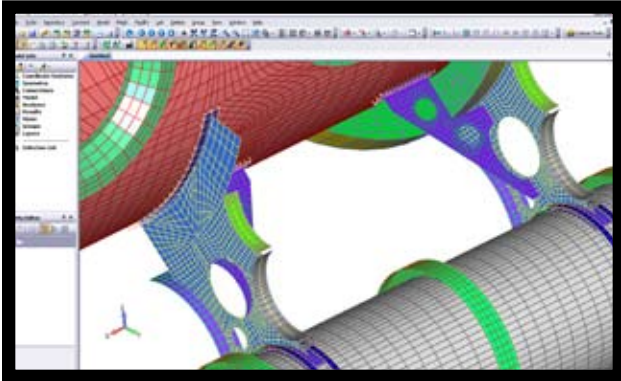

Image courtesy of Predictive Engineering, Inc.

On the CAD side of the workflow, Femap will both import and export DXF and IGES (points and curves), STL, ACIS (.sat) Parasolid (parts or assemblies). It will also import data from CATIA (v4.1.x, v4.2, or v5), VDA, I-deas, Pro/Engineer, Solid Edge and NX. New for the Femap 10 release is the ability to import Parasolid data directly from SolidWorks parts and assemblies, which means, as Femap is also Parasolid-based, you'll get a direct transfer of geometry.
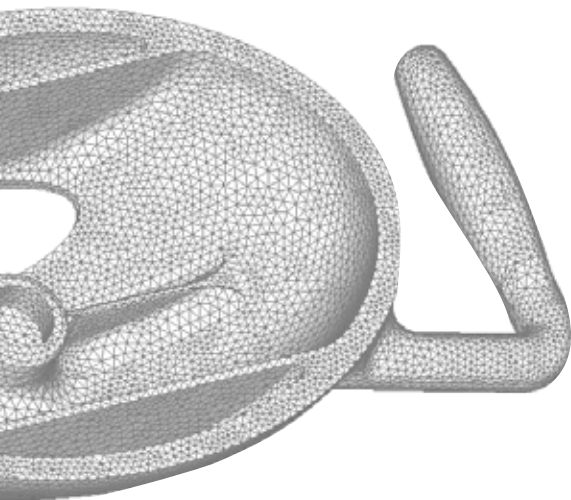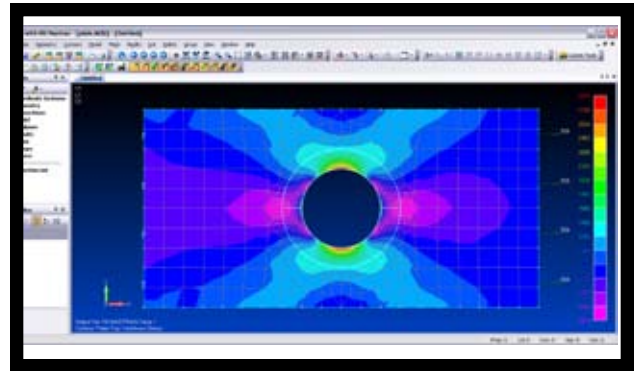

Image courtesy of Predictive Engineering, Inc.

When it comes to simulation or analysis solver connectivity, Femap can work out-of-the-box with NX Nastran as well as other flavours of Nastran. In house or heavily customized code is also handled well, with Femap's well-developed and comprehensive API for custom programming.

As a conduit between the CAD geometry authoring systems and the simulation solver code, Femap provides the user with a huge array of tools to cleanup and repurpose geometry for for the simulation process. Ths includes tools to create, correct and improve meshes and loading conditions to suit the purposes of simulation and the products performance conditions as well as a suite of results viewing and investigation functionality that allows you to make and document informed decisions based on that simulation. With Femap 10, there has been work done on the system all across the board, so let's dive in and see what it can do.

"Femap version 10... brings automation to the fore, but does so with a series of tools that allow the user to adapt and refine the mesh and loading conditions... to ensure results are accurate...."

Intelligent auto-meshing

Geometry is key for any simulation task, but there is often a disconnect between what the analysis focussed user desires and that produced by the authoring 3D design system. While a 3D model might be valid in a CAD system, for the purposes of effective and accurate simulation, it might be less than perfect. A core update for this release is a series of meshing technologies that change how Femap creates a mesh based on geometry, both complex surface-based as well as mixed meshes that we'll get onto shortly.

The concept is that instead of using the parameters defined by the surface geometry of a 3D model for meshing, Femap will create meshes in 3D space. While that sounds incredibly vague, the point is that Femap creates that critical mesh on your part or assembly as a whole, taking into account different element types, and the requirements of simulation, rather than tying it to face boundaries. As a first pass mesh, this will get you much closer to an ideal mesh because the system is creating the mesh on the basis of what is best for simulation, rather than trying to work around CAD geometry.

While the auto-meshing tools will give you a higher quality mesh in many cases, when you start to deal with complex or less than perfect geometry, you'll always need to be able to dive in and adapt the mesh manually. What's then needed is a series of tools that allow you to dive in and make further adjustments to the mesh, to control how it's defined and used in simulation to ensure that it's perfectly suited to your requirements – and the good news is that this has been built directly into Femap with the Meshing Toolbox. So let's look at that.
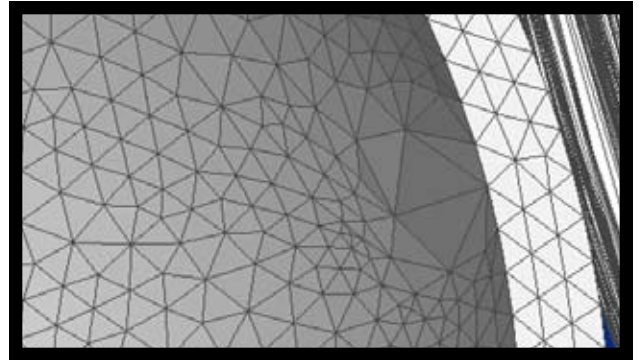
When face geometry is used to define element size, common problems are poles such as these shown above which can cause errors in your results due to poor mesh creation.
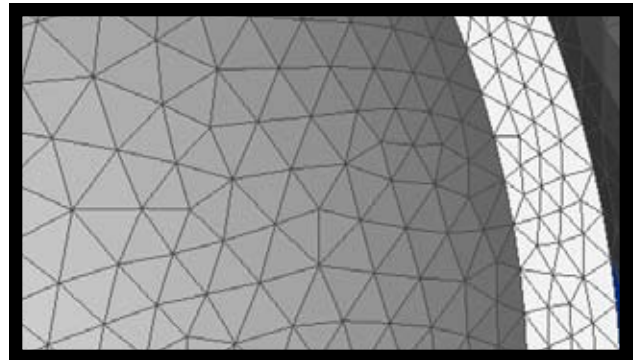
The new automated meshing tools use 3D space, as opposed to explicit face geometry, as the basis of the mesh creation. As a result the first pass mesh is typically much higher quality.
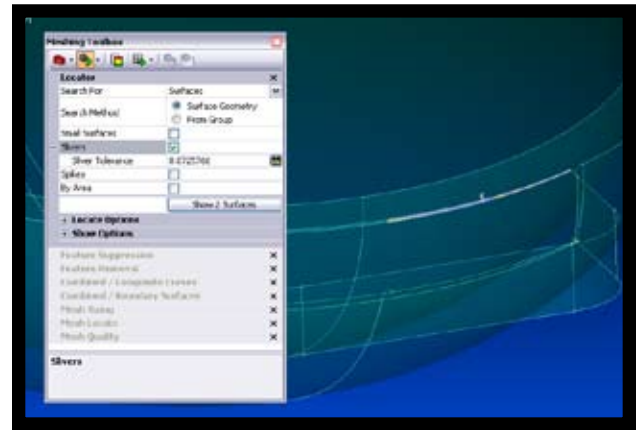
Meshing Toolbox

The Meshing Toolbox brings together all of the pre-existing tools for geometry clean up and mesh refinement with a series of new tools that both add functionality and make the process more usable. The deceptively small and compact dialog is laid out in a pretty linear manner, presenting a series of tools that are typically used in the same order, so let's step through them and look at what each does. The first challenge is finding those errors such as short edges, sliver faces and the like.

While some problems are immediately obvious to the trained eye; many are not, so a new tool, the Entity Locator, comes into its own. Built into the Meshing Toolbox, it presents a quick way to find problematic geometry conditions such as short, free and non-manifold edges, sliver surfaces and other geometry that can wreak havoc with your meshing process. These are typically very small entities and having a tool that can find them, using user controlled filters (such as size, by group, by location), enables you to find them and deal with them. Now, let's look at how you go about solving these issues.

The first two commands in the Toolbox are Feature Suppression and Feature removal. Feature Suppression is used to identify and mask out geometry features (such as small loops, curves and surfaces) that are not necessarily required for the study at hand, but might be required later on in further studies. Essentially, by suppressing a feature (such as small holes), the system will ignore them during meshing routines but retain that geometry in case it's required downstream. Feature Removal works with the same features, but removes them completely.

The choice of which operation is used depends entirely on your requirements and specific study, but as a default, it's perhaps best to use the suppression option, as this means that data is retained for future use.

Meshing Toolbox allows you to find short edges, sliver faces, then provides a range of commands for dealing with those common issues.

The next pair of tools deal with combining both problematic curves and surfaces. It's often the case that the authoring CAD system will create very small curves, badly constructed (in terms of meshing) curves that need to be concatenated to create a much simpler form. And the same holds true of surfaces. While these are perfectly valid for CAD purposes, again they can cause problems with meshing algorithms which typically deal with faces (in terms of creating the same number of elements within that boundary) whether they are large or small – so you either get unwanted or ill-advised concentration of elements in those areas or the mesh simply isn't created. You use the entity locator to find such small curves and faces, then use the tools to combine them to create a more sensible set of geometry.

Geometry editing

The Mesh Toolbox includes tools for Geometry Cleanup, but what happens if you need to make more dramatic changes to your geometry before you start meshing? There are many instances where the only way you can get close to an ideal mesh is to dive in and manually adapt either the underlying geometry or the mesh itself. Being a pre-processor, Femap includes these tools too and they've seen some work for the 10th release which makes them more usable by adding common used functions for editing geometry, but also in supporting commonly conducted workflows.

The new Geometry Splitters allow you to subdivide surface geometry using a number of different methods, which depend on both the result you have and the geometry conditions you've encountered. You can create a basic split by simply selecting two points on edges to split it between. You can also reference other geometry in surrounding surfaces, such as edges from which you might want to continue tangency. The other two tools allow you to maintain tangency from one edge to a point or between two edges.

Alongside the splitting tools are two intriguing little commands; Washer and Pad. These automate commonly carried out tasks that an analyst would do on a very regular basis to refine the mesh around specific features such as stress raisers, whether that's in the form of an offset from an arbitrary curve, around a circular hole or a pad.

There are explicit commands for each and they make very light work of these tasks. You select the geometry you want to work around, add the options (for example, for the washer, whether

to split the washer or not) and away you go. As you can see from the examples shown, its painless to create these features that allow you to concentrate mesh elements where needed and to handle stress raisers, using mapped meshing methods without adversely overburdening the complete mesh. When you're working with larger scale parts, the use of these tools can also bring order to unordered meshes, by providing meshing guidance.
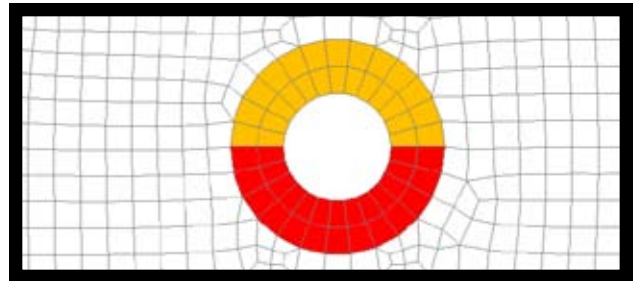
Curve Washer is used to create an orderly mesh around a stress raiser
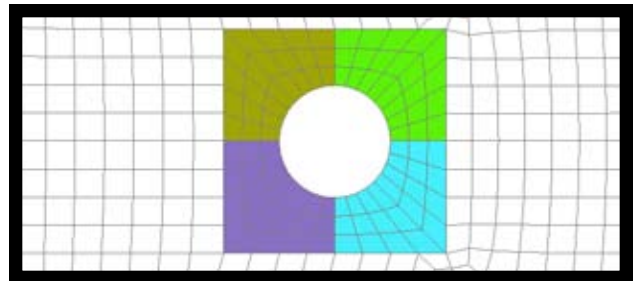
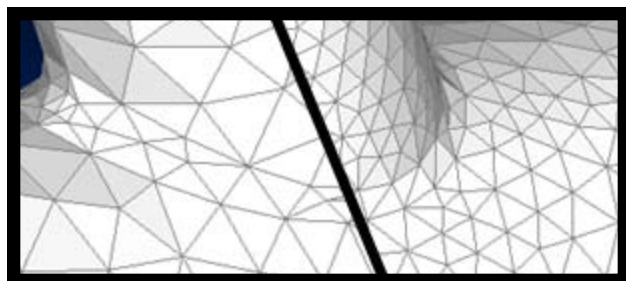*Curve Pad is used to create an orderly mesh around a stress raiser*
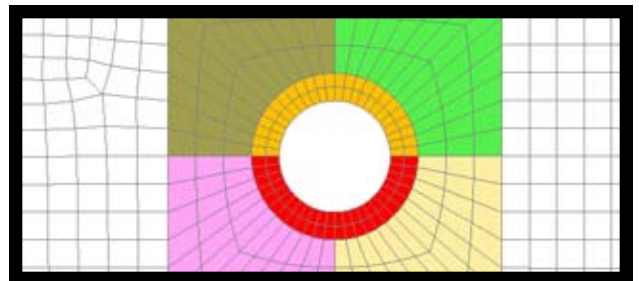
Curve Washer and Pad in combination are used to create an orderly mesh around a stress raiser
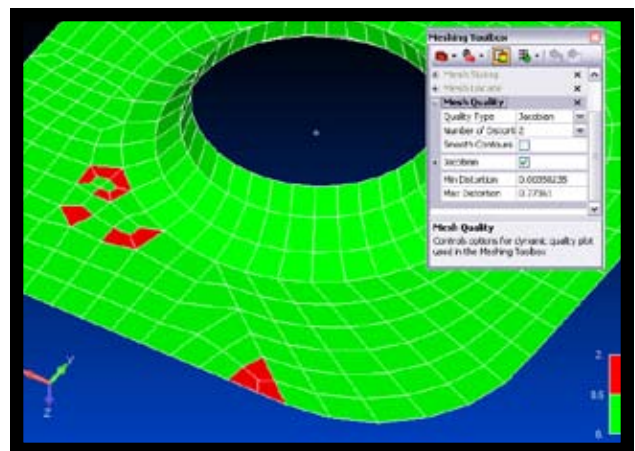
Interactive mesh editing

Once you have your geometry in a fit state, its time to create your mesh. While the new meshing routines we've already discussed will get you so far, as ever, the details are what's going to make the difference between a good mesh that works and a mesh that suits its purpose perfectly – and that's always going involve some form of editing, of refinement and adaptation. To assist with this incredibly detailed and traditionally lengthy process, Femap 10 sees the introduction of some new tools, interactively, dynamically and accurately.

In the first instance and probably as a first pass, you'll find areas where the mesh needs to be quickly refined, either by increasing the element count in areas (where stresses concentrate) or non-critical areas where fewer elements can be used for the sake of efficiency. Using the Mesh Sizing tools from the Meshing Toolbox, you can do this by simply adding or subtracting element counts. Adding means that clicking an edge will subdivide that into a greater number of nodes to increase element count, while subtracting removes them. You have complete control over how this is done from within the Toolbox and as you work on areas of interest, the mesh updates instantly and you quickly work around areas that need work.

Interactive meshing uses an interactive 'painting' method to add or remove elements. Mesh before (left) and after adding elements (right)

The next stage of refinement is more granular, as rather than refining the mesh in large sweeps, its common that you need to work on specific individual or small clusters of problematic elements. The Mesh editing tools allow you to find these and work on each node. Again, this is dynamically driven, you grab the node you want to shift, select whether you are moving it along the inherent curve or on the surface of the mesh or indeed, if you're feeling brave, in free space.


Whichever of these tools is being used (chances are that you'll use both), another new addition to this release is the Live Quality Checking tool (shown above), which displays graphically the quality of the mesh, down to an individual element level. The mesh and elements can be color coded by quality, according to fully customisable criteria in terms of aspect ratio, internal angle, warp, taper and Jacobian. As you refine the mesh or move nodes, the whole display updates live so the effect of your edits can be evaluated, not only on the element group you're working on, but the mesh as a whole. Incidentally, the NX Nastran quality checks have also been added, so equivalent quality checks can now be viewed in Femap prior to analysis.

7

Mixed meshing updates

Alongside the steps we've already covered, another core focus for Femap 10 is the handling of both surface shells with non-manifold geometry and mixed meshing. With today's processing power, its often tempting to simply tetrahedral mesh everything, but when you break down a component, you'll typically find that selective use of other meshing techniques can pay dividends – something that the analyst community has been aware of for many years.

By combining common-or-garden tetrahedral meshing with beam, shell and mid-surface elements, you can build a study that solves in a much shorter space of time with the same accurate results. While that's the ideal case, those with experience know that mixing such geometry and meshing methods can be problematic, particularly where two distinct and individual groups of elements meet. In the first instance, when using mid-plane or shell elements gaps occur where the interface between them is less than ideal.
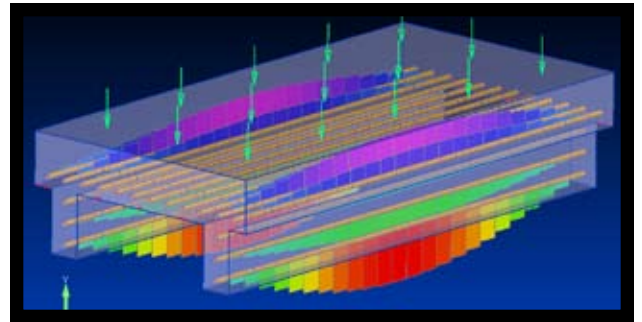
Femap ensures a contiguous mesh for mixed element models such as these reinforcement beam elements inside a solid.

Femap now includes tools that can connect these up completely by extending geometry and creating a manifold. Over and above this, the system now also includes tools for adapting the mesh in these critical junctions, to ensure that elements and nodes match up in a cohesive manner, so the mesh is cleanly laid out and forces can be transferred between the element types without any concern. When combined with the tools for improving element concentration at stress raisers you have a very powerful way to quickly create an effective mesh quickly.

Mid-surface modeling tool

We've already discussed the new tools for working with mixed meshing methods within a single simulation study, but often, to get to that point, you will need to calculate a mid-surface model from your geometry – which makes the simulation of uniform thickness parts much more efficient. While this would have traditionally been somewhat painful, this new command automates their creation. You simply select a face either on the 'inside' or 'outside' of the model, Femap finds tangentially connected surfaces (within a defined tolerance for less than perfect geometry) and creates the mid-surface between the inner and outer limits. While this is initially about creating a more efficient study (by using shell elements), it also enables you to use that common mid-surface model in optimization processes, where you can find the best weight or material trade off against structural strength, by simply adapting the thickness value and rerunning the study, rather than having to remesh at different material thicknesses for each iteration.

Updates for NX Nastran

Since Siemens PLM Software acquired the rights to develop and distribute its own branded version of the Nastran solver, the links between NX Nastran and Femap have been improved with each release and for those using both product in concert, this Femap 10 release see a number of updates that will make the process more efficient.

On the meshing side of things, there are new axisymmetric elements (quads and tets) that match between the two systems. There's also two new elements for those working with welded or bolted connections in their products: the new CWELD extensions for patching in weld elements while the new CFAST allows you to do the same with Fasteners. These match the Nastran elements exactly, so you can set-up the study in a single system and send it to solve, without intermediary steps.

For those working with various data sources, the ability to reconnect your mesh and the underlying geometry is going to make life easier. It may be that you load the geometry and the finished mesh into Femap separately from another system (Patran for example). In such a case Femap 10 can automatically re-establish the link between the two.

Finally, Femap is used as the pre/postprocessor for NX Nastran so its ability to monitor NX Nastran as it solves is pretty critical. The iterative nature of simulation means that being able to spot problems associated with non-convergence of results, errors in your process, directly from the tool you use to make changes to the mesh, the boundary conditions and such, means that the process works more efficiently from a single window.
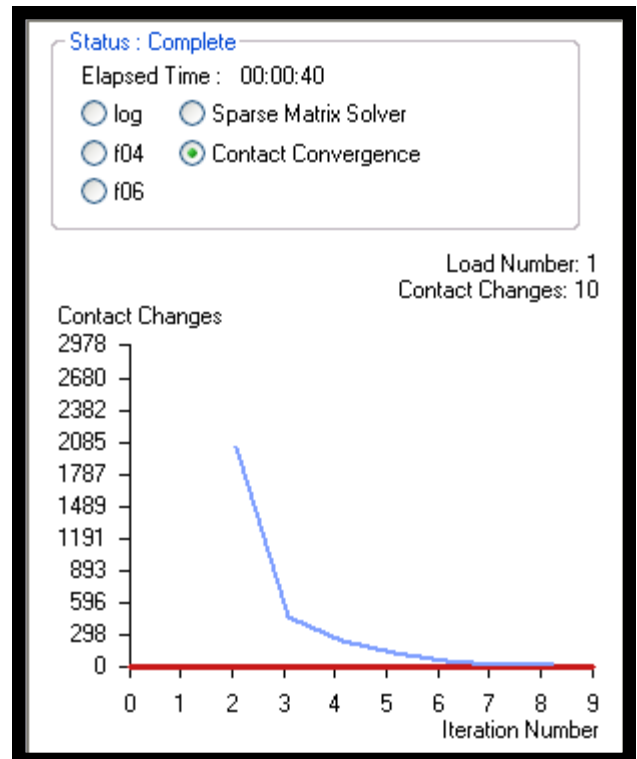
The analysis monitor shows a graphical display of the progress of the analysis while it's running

Results visualization updates

Big news for Femap 10 is the ability to transform your results display, which makes it easier to switch between different coordinate systems. This gives you the ability to match your results display to the load or operating condition that you're simulating, in a very explicit manner. The system already includes a huge raft of result visualization methods and filters and this backs these up.
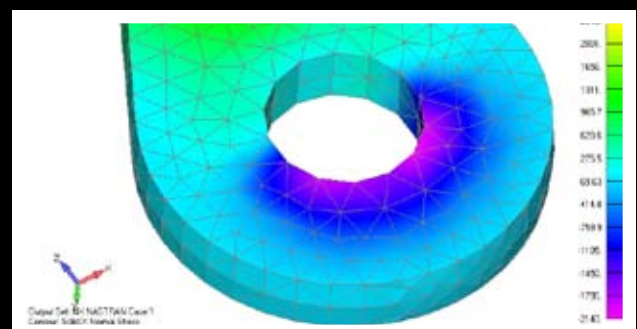
Conclusion

As we discussed at the outset, Femap has built a reputation over the years for being the Swiss Army knife for simulation. By focussing on the pre and post processing workflows and integration to a wide range of solver codes, the system can sit between almost any solver currently in use (commercial or even in-house). What this has also done is given the development team exposure to a much wider range of problems, issues and workflow bottlenecks that arise within the simulation community. And with that knowledge and some talented and experienced developers, you get a release such as Femap 10 that brings a huge range of new tools to the fore. Looking at the advances made with this release alone, perhaps the best way to summarize the Femap 10 release is that it's all about one thing: Getting the job done.

To take that further, the goal of any software within the professional and technical realm should be to provide tools that support and enhance the user's knowledge and expertise. In the case of pre-processing for simulation, that knowledge pertains to knowing what constitutes a good mesh. Femap 10's goal is let those users use that knowledge in a highly efficient manner. This is accomplished in part by providing levels of automation both where needed and suitable, such as the new 3D space-based meshing routines, which get you much closer to a good mesh on the first pass. Once you've have that first pass mesh, tools are available that let you dive in, find potential areas of problems and fix them.

With each Femap release these tools get more efficient and more intelligent, so the process becomes much easier. Femap 10 provides intelligence (in terms of process coverage) backed up with dynamic feedback and interactivity. This lets you refine your mesh in a shorter space of time, because not only do you work on it directly, but you also receive a visual guide to the effects of your modifications. Whether that's combining curves, creating composite surfaces or moving individual nodes. All this adds up the fact that the job, as we said was the desirable end result, gets done in a shorter timeframe.

But how does all this tie into the business of developing products? As said at the outset, if you're going to take advantage of digital design tools, then you need to ensure that design and simulation iterate quickly and in synchronisation. Make a design change, validate it digitally, and then make further design changes; that's the cycle. What Femap does is provide an environment in which that validation can take place efficiently and accurately. It can integrate with all the leading technologies out there, in terms of both 3D authoring systems and solver codes and with today's user typically not restricting themselves to just one vendor, the benefits of having a single interface through which all manner of simulation activities can take place, makes even more sense.

www.siemens.com/plm